What's New in Mastercam X8

July 2014
**Mastercam® X8 What's New**

Date: July 2014  
Copyright © 2014 CNC Software, Inc.— All rights reserved.  
Software: Mastercam X8

**TERMS OF USE**  
Use of this document is subject to the Mastercam End User License Agreement. A copy of the Mastercam End User License Agreement is included with the Mastercam product package of which this document is part. The Mastercam End User License Agreement can also be found at:  
[http://www.mastercam.com/companyinfo/legal/LicenseAgreement.aspx](http://www.mastercam.com/companyinfo/legal/LicenseAgreement.aspx)

**Be sure you have the latest information!**  
Information might have been changed or added since this document was published. The latest version of this document is installed with Mastercam or can be obtained from your local Reseller. A ReadMe file (ReadMe.pdf)—installed with each release—includes the latest information about Mastercam features and enhancements.
Contents

Introduction.................................................................................................................. 1
  ▶ Mastercam Documentation...................................................................................... 1
  ▶ Contact Us .............................................................................................................. 1
  ▶ Mastercam Resources ............................................................................................ 1

System Enhancements................................................................................................. 3
  ▶ Graphics Upgrades ................................................................................................ 3
  ▶ New Customize Interface ....................................................................................... 3
  ▶ Mastercam Simulator Improvements ................................................................... 4
    Viewports ................................................................................................................ 4
    Part Sectioning ........................................................................................................ 5
    Drill Cycle Handling ............................................................................................... 7
    Workpiece Tolerance ................................................................................................ 8
    Exporting Support Data .......................................................................................... 9
    Additional Simulator Improvements ....................................................................... 9
  ▶ Code Expert Editor Improvements ..................................................................... 10
    Multi-Stream Navigation ......................................................................................... 10
    NC Configuration Enhancements ......................................................................... 10
    New Find Extents Functionality ............................................................................ 10
    Additional Enhancements ...................................................................................... 10
  ▶ Plane Improvements ............................................................................................. 10
  ▶ System Configuration Enhancements .................................................................. 13
  ▶ File Management Improvements ......................................................................... 14
  ▶ Graphics Interaction Improvements ..................................................................... 14
  ▶ File Translators ....................................................................................................... 15
  ▶ Migration Wizard .................................................................................................. 16
**Design Enhancements**

- New Solids Workflow ......................................................... 17
- New Interface ........................................................................... 17
- Live Preview .............................................................................. 17
- OK/New .................................................................................. 17
- OK/Regen ................................................................................ 17
- Improved Selection .................................................................... 18
- Hands-on Drag Controls .......................................................... 18
- Non-interrupting Error Messages ........................................... 18
- History Tree ............................................................................. 19
- **Solid Direct Editing** ........................................................... 19
  - Push/Pull ............................................................................... 19
  - Move ..................................................................................... 19
  - Split ..................................................................................... 19
  - Simplify ................................................................................ 19
- **Smart Chains in Solids** ....................................................... 20
- **Primitives Default to Solids** ............................................... 20
- **Solid Edge Awareness** ......................................................... 20
- **Surface Modeling Improvements** ....................................... 23
  - Change Normal ..................................................................... 23
  - Set Normal ........................................................................... 23
  - Orient Normals ..................................................................... 23
  - Surface to Surface and Surface to Plane Auto Analyze ........ 23

**Mill Enhancements**

- **2D Milling** ........................................................................... 25
  - 2D HST Improvements .......................................................... 25
  - Region Chaining ................................................................. 26
  - 2D Contour Enhancements ................................................... 30
- **3D Milling** ........................................................................... 30
  - Z Stepdowns ........................................................................ 31
  - 3D HST Project Depth Cuts .................................................. 32
Tilt Away to Avoid Collision ................................................................. 32
  ▶ Multiaxis Milling ................................................................. 34
  ▶ Multiaxis Roughing .............................................................. 34
  ▶ Safety Zone Improvements .................................................... 34
  ▶ Tool Inspection ................................................................. 35
  ▶ Additional Improvements ..................................................... 36
  ▶ Mill Tooling ................................................................. 36
  ▶ Tool Creation and Editing ..................................................... 36
  ▶ Edit Holder ................................................................. 37
  ▶ Edit Assembly Names ........................................................ 38
  ▶ Calculate Feeds and Speeds .................................................. 40
  ▶ Standalone Tool Manager .................................................... 41
  ▶ Productivity + Enhancements ................................................ 41

**Lathe Enhancements** ................................................................. 43
  ▶ Lathe Stock Model Support ..................................................... 43
  ▶ New Contour Roughing Toolpath .......................................... 44
  ▶ Additional Improvements ..................................................... 44

**Mill-Turn Enhancements** ............................................................ 45
  ▶ Level 1 Now Available ........................................................ 45
  ▶ Reference Positions ............................................................ 45
  ▶ Wibu Licensing ................................................................. 46
  ▶ Job Setup Enhancements ....................................................... 46
  ▶ Tool Loading Enhancements ................................................ 48
  ▶ Sync Manager Enhancements ............................................... 48
  ▶ Machine Property Changes .................................................. 49
  ▶ Additional Enhancements ..................................................... 49
Introduction

Welcome to Mastercam X8! Mastercam X8 features new products and new functionality focused on delivering speed and efficiency for your machining jobs. We are sure that you will benefit from what Mastercam X8 has to offer you and your shop.

Mastercam Documentation

Mastercam installs the following documents in the \Documentation folder of your Mastercam installation:

- What's New in Mastercam X8
- Mastercam X8 Installation Guide
- Mastercam X8 Administrator Guide
- Mastercam X8 Transition Guide
- Mastercam X8 Quick Reference Card
- Mastercam X8 Post Debugger User's Guide
- Getting Started with Renishaw Productivity+™
- Mastercam X8 ReadMe

Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- Mastercam Help—Access Mastercam Help by selecting Help, Contents from Mastercam's menu bar or by pressing [Alt+H] on your keyboard. Also, most dialog boxes and ribbon bars feature a Help button that opens Mastercam Help directly to related information.
- Mastercam Reseller—Your local Mastercam Reseller can help with most questions about Mastercam.
- Technical Support—CNC Software's Technical Support department (860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
Mastercam University—CNC Software sponsors Mastercam University, an affordable online learning platform that gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master your skills at your own pace and help prepare yourself for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email training@mastercam.com.

Online communities—You can find a wealth of information, including many videos, at www.mastercam.com. For tech tips and the latest Mastercam news, you can join us on Facebook (www.facebook.com/mastercam), follow us on Twitter (www.twitter.com/mastercam), and subscribe to our blog, Mastercam Xtras (http://blog.mastercam.com). Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)!

Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the knowledge base at kb.mastercam.com. To register, select Help, Link on Mastercam.com from the Mastercam menu and follow the instructions.
System Enhancements

**Graphics Upgrades**

The graphics display in Mastercam X8 has been upgraded to provide cleaner aesthetics, along with clearer and more responsive selection while delivering a much improved overall look and feel. These improvements include:

- Enhanced model rendering with sharp edges and optimal lighting.
- New highlighting methods for model and chain selection make selection faster, clearer, and easier.
- New default gradient background and background colors.
- New modern color palette for toolpaths, wireframe geometry, surfaces, solids, etc.
- Individual color settings for solids, surfaces, and wireframe geometry.
- New **Solid edge color** setting.
- New **Surface edge color** setting.

**New Customize Interface**

Mastercam X8 uses workspace files to customize your Mastercam look and feel. *Workspace* files combine key mapping, toolbar states, screen size and location, right-click menu settings, and other interface options. Choose **Settings, Customize** to experience this improved look and feel.
Mastercam Simulator Improvements

Viewports

A new View ribbon bar has been added for both Backplot and Verify. The ribbon bar contains some items from the Home page (Show and 3D View) and adds two new sections called Window and Viewports. The Viewport tools are a new feature to both Backplot and Verify that split the graphics window into one of four configurations. Each viewport offers separate view orientation and control. Right-click options only work in the current view (outlined with a red border). Visibility options affect all viewports (workpiece display, tool display, etc.). The Restore default view option resets
the configuration back to the original size. You can save the viewport configuration to your default settings by choosing **Home, Save to Defaults**.

---

**Part Sectioning**

The view sectioning tool on the Verify ribbon bar has been expanded to add three additional plane orientations. In an improvement targeted toward Mill users, when the planes are initially displayed, they are oriented around the center of gravity of your part. Even if your part is not near the system origin, it will still section efficiently.
Selecting one of the ¼ view orientations displays two shaded planes - these clipping planes can be moved by left-click/hold and dragging the cursor to a new location.
You can also use just one clipping plane at a time to create the part view you need.

**Drill Cycle Handling**

Drill cycles in both Backplot and Verify now respect the settings in the parameter pages, so you see the tool moving in and out as defined. This applies to all cycles for Mill and Lathe.
Workpiece Tolerance

The Backplot/Verify Options dialog box includes a new **Workpiece tolerance** option. Targeted towards users with very small parts, it gives you better control over the workpiece quality. The Verify/Compare feature provides better results, since you can now match the tolerance between the part and the workpiece.
Exporting Support Data

To improve customer support, you can now save your Mastercam Simulator session data to a file to send to CNC Software, Inc. for analysis. Choose **File, Help, Export Support Data**. Enter a file name and all the data for your Simulator session is saved.

Additional Simulator Improvements

- Improved image quality of custom (STL or solid) stock.
- Display correct feed rate for each move.
- Corrected displayed values in the Move List and measuring tools, which are now relative to the current toolpath’s WCS instead of world coordinates.
- Included support for Steady Rest operation.
- Improved lighting source and direction.
- Added message when Stop Condition is triggered.
- Added support for stock origin with WCS offsets.
- Improved quality and reduced size of mesh saved in STL file.
**Code Expert Editor Improvements**

**Multi-Stream Navigation**

The editor in Mastercam Code Expert now properly supports the display of any number of streams. The editor uses the stream definition in NC Configuration and the starting and ending characters for each stream to determine if single or multiple files are required. You can toggle the display of each stream on and off in a new Navigation pane. Sync codes are now graphically linked between visible streams, and the Navigator shows which streams contain associated sync points when you hover over the code.

**NC Configuration Enhancements**

NC Configuration allows you to tailor Code Expert display for NC files by defining the number of streams, file extensions, sync characters, tool callouts, comments, and other features for individual machine output.

You can access the new interface for creating and modifying the NC Configurations in the **Application Options** dialog box and directly using the NC Document ribbon bar.

Default NC Configurations are provided for several machine and control types, providing easy starting points for user customization.

**New Find Extents Functionality**

NC File document types now include an Extents pane which lists the extents for address characters in the NC file. The address characters are easily defined for reuse in the NC Configuration and can be customized in each NC document type. Double-clicking on an extent moves the cursor in the NC document to the location where the value appears.

**Additional Enhancements**

- Fonts and Colors are now fully customizable for each document type.
- **Go To Bookmark** functionality added, which allows you to add a bookmark to a document and easily return to that bookmark as desired.

**Plane Improvements**

Throughout Mastercam X8, the term “view” has been replaced with the term “plane” (except for graphics views). A Mastercam Plane contains a view and an origin, and a
View is a matrix defining an orientation. In keeping with this change, the View Manager is now the Plane Manager.
New options are also available in the right-click menu of this dialog box.

- Planes are marked “dirty” if their regeneration fails. Any toolpaths that use that plane will also be marked dirty.
- You cannot make changes to the default planes. You also cannot delete a plane that fits any of the following criteria:
  - Plane is locked.
  - Plane is being used as the active Tplane, Cplane, WCS, or Gview.
  - Plane is used in a machine group.
- Plane names are no longer limited to 40 characters.
- Selecting geometry to define a plane will always offer eight possible plane options.
- The WCS, Tplane, and Cplane names have been moved to the status bar to provide a cleaner drawing space. When you hover over the status bar area, a
tooltip displays containing extended plane information. The Gview (if defined) remains on the screen.

- Automatic work offset assignment has been improved, with better performance and handling of conflicting assignments. In addition, the Work offset already used warning can now be turned off.

**System Configuration Enhancements**

- The Large Buttons setting has been moved from Configuration to Customize.
- Multiple viewport support and GDI graphics support options have been removed.
- The Toolbars and Keymap startup settings have been replaced by a Workspace startup setting.
Complicated lighting settings have been replaced with a new Lighting mode setting. The session-only Shading Settings command and dialog have also been removed.

“Mastercam” communication functionality has been removed. CIMCO 7 is now the default and only distributed option.

File Management Improvements

You can now open all supported file types by dragging files from Windows Explorer or your desktop and dropping them onto the graphics area of an open Mastercam session. You can also drag and drop a file onto the Desktop shortcut to open a new session of Mastercam.

In addition, uppercase and lowercase characters are now supported for file names, file extensions, plane names, etc. and are no longer forced to all uppercase.

Graphics Interaction Improvements

The 3D Gnomon has been modified and is used as the primary control in the new Move function along with Dynamic Xform and Dynamic Planes. When
using the linear or rotate options, an editable field displays the current values and allows you to directly input values.

- Pressing [Spacebar] now repeats the last command when in an idle state.
- The middle mouse button modifier key to toggle panning and spinning has been changed from [Alt] to [Shift].
- The key used to dynamically rotate about the screen normal has been changed from [Shift] to [Ctrl].
- Holding down the X, Y, or Z key during dynamic rotation now locks the respective axis. This requires the Use ‘Free’ mode in Dynamic Spin option to be on.
- Toolpaths now remain visible during dynamic rotation.

**File Translators**

- File importers which import solids will now name the solid to match the file name.
- Catia V5 - added support for V5 R23.
- Unigraphics - added import support for NX9.
- Parasolids - added support for V26.
Migration Wizard

Mastercam’s Update Folder and Migration Utility features have been combined into a new Migration Wizard. You can access this new feature by choosing File, Migration Wizard.

The Migration Wizard offers a Basic mode and an Advanced mode. The Basic mode will automatically update all Mastercam file types of the previous version (for example, Mastercam X7) from a given location, while the Advanced mode provides additional control over file types and previous versions to migrate.

The file update process has also been improved to provide a seamless, more efficient experience. Migration results are now displayed in the Mastercam Event Log as well.
Design Enhancements

*New Solids Workflow*

The solids interface and workflow has been completely redone for Mastercam X8 and makes constructing and editing Solids a lot easier and more intuitive than it’s ever been. Every existing Solids function has been updated to reflect this new workflow.

*New Interface*

All solids functions have been given a new modern interface that allows you to work in the interface and on the screen as needed.

*Live Preview*

With the live preview, you will now see the results of changes made in the interface as you work - no more entering values into a dialog box and then clicking OK hoping you didn’t make a mistake.

*OK/New*

While creating a new operation, you have the option of clicking OK to finish the operation and exit the function, or clicking OK/New which will finish the operation and start a new one without forcing you to restart the function manually.

*OK/Regen*

Prior to this enhancement, any changes made to an existing operation required you to click the Regen button at the top of the Solids Manager. In X8, we added an OK/Regen button that exits the function and regenerates the solid at the same time.
Improved Selection

By putting the solid selection filters in a new dialog that pops up at the start of an operation, it’s easier to set them up as you are working.

Holding the [Shift] key down while selecting an edge selects all tangent edges at the same time, and holding [Shift] while selecting a face will select all tangent faces at the same time. Wireframe, faces, edges, and bodies can be edited inside of the dialog itself. This eliminates the need for a separate branch in the Solid tree and makes adding and removing items much easier.

Hands-on Drag Controls

The single-axis arrow is used in various functions in X8 Solids and can be used to set distances on the screen without having to enter them manually in the dialog. This includes the ability to type in values on the fly when those controls are in use. The new Push/Pull functionality relies on it (see page 19 for more information), and it supplements the new interface of Extrude, Sweep, Revolve, Thicken, and Shell.

Non-interrupting Error Messages

With the new Live preview capability, most error messages have been made so they don’t interrupt the workflow. A warning text box displays in the upper right corner of the screen to show that the solid in its current state of editing has an error. The message may just be a warning or may be more informative. This allows you to make
changes to eliminate this error state without having to stop what you are doing and dismiss a pop-up dialog.

**History Tree**
In earlier releases of Mastercam Solids, the creation and editing of operations had different rules and workflows. Now editing and creating operations is approached in the same manner, which simplifies the Solid History and reduces each operation to a single branch instead of three branches. In X8, all you need to do is double-click on an operation to open it for editing.

**Solid Direct Editing**
In X7, we began to work in the area of direct editing solid bodies. Many users import solids from other systems and have no easy way to prepare these models for machining. In X8, we have greatly expanded these capabilities to make the process of model preparation easier. All of these options have been placed under a new **Model Prep** menu.

**NOTE:** All of our direct editing functions remove the solid history. **Undo** restores the history as it was before the direct editing function.

**Push/Pull**
Use it to offset faces or push edges into fillets.

**Move**
The Move function works with faces and allows you to translate or rotate them on your solid body. It can be used to move or copy faces or even just to add draft.

**Split**
The Split function in X8 splits a solid face into multiple faces to be used with Move or Push/Pull. Split supports two methods: Wireframe and Flowline. The Wireframe method projects wireframe geometry onto a face to form new edges and faces. The Flowline or UV method splits a face along its UV flowlines.

**Simplify**
Simplify was introduced in X7 but has been improved in X8. This function simplifies a body and combines faces on a case-by-case basis by selecting faces or edges.
Smart Chains in Solids

This enhancement automatically reconnects wireframe chains that have been edited so you don't have to re-chain them. The classic example of the smart chain enhancement is seen when you extrude a rectangle, then go back and fillet a corner. In previous versions, you would have to re-chain the wireframe; in X8, you just have to regenerate the operation. This enhancement affects Extrude, Revolve, and Sweep operations.

Primitives Default to Solids

On systems with Solids enabled, all primitives will default to solids instead of surfaces. If you manually switch to surfaces, then the change will remain modal for the session.

Solid Edge Awareness

Back in X6, we introduced the ability to use solid edges for trimming wireframe entities to solid edges and to find theoretical intersections between solid edges in space. Both changes were aimed at eliminating the need to create wireframe as a reference entity where a solid edge already exists.

For X8, we have taken this much further and adapted the idea of Solid Edge Awareness throughout wireframe modeling. Our goal for X8 is to allow solid edges to be used as reference entities and eliminate the need to create temporary geometry.

In case you want to ignore a solid edge and force a function to pick wireframe instead, we tied Solid Edge Awareness to the Edge filter on the General Selection bar. If you are in a function that supports Solid Edge Awareness, the Edge filter will be available and can be used to recognize or ignore solid edges for that function.

The following images show the three states the General Selection bar will use to indicate if a function is Solid Edge aware and if it is currently active or not.

This is how the General Selection bar looks if Solid Edges are not supported in the current function:

This is how the General Selection bar looks if Solid Edges are supported, but not enabled for the current function (note the solids icons):
This is how the General Selection bar looks if Solid Edges are supported and enabled for the current function:

Here is a current list of functions that support Solid Edges as reference geometry:

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AutoCursor</td>
<td>Relative Along an Entity</td>
</tr>
<tr>
<td></td>
<td>Nearest</td>
</tr>
<tr>
<td></td>
<td>Perpendicular from a position to an edge</td>
</tr>
<tr>
<td></td>
<td>Tangent from a position to an edge</td>
</tr>
<tr>
<td></td>
<td>Intersection</td>
</tr>
<tr>
<td>Trim/Divide</td>
<td>*Solid edges are not edited but can be used</td>
</tr>
<tr>
<td></td>
<td>Trim 1</td>
</tr>
<tr>
<td></td>
<td>Trim 2</td>
</tr>
<tr>
<td></td>
<td>Trim 3</td>
</tr>
<tr>
<td></td>
<td>Divide</td>
</tr>
<tr>
<td>Break at intersection</td>
<td>(used as an intersection entity)</td>
</tr>
<tr>
<td>Xform</td>
<td>Offset (single entity offset)</td>
</tr>
<tr>
<td></td>
<td>Mirror</td>
</tr>
<tr>
<td>Dimensions</td>
<td>Linear</td>
</tr>
<tr>
<td></td>
<td>Angular</td>
</tr>
<tr>
<td>Points</td>
<td>Dynamic</td>
</tr>
</tbody>
</table>
### Segment

<table>
<thead>
<tr>
<th>Line</th>
<th>Parallel</th>
<th>Closest</th>
<th>Bisect</th>
<th>Perpendicular</th>
<th>Tangent through a point</th>
</tr>
</thead>
</table>

### Arcs and Circles

<table>
<thead>
<tr>
<th>Arcs and Circles</th>
<th>Circle Center Point - for tangency conditions</th>
<th>Arc Polar - for tangency conditions</th>
<th>Circle Edge Point</th>
<th>Arc Endpoints</th>
<th>Arc 3 points - tangency conditions</th>
<th>Arc Tangent - 1 entity</th>
<th>Arc Tangent - point</th>
<th>Arc Tangent - Center Line</th>
<th>Arc Tangent - Dynamic Tangency</th>
<th>Arc Tangent - 3 Entities</th>
<th>Arc Tangent - 2 Entities</th>
</tr>
</thead>
</table>

### Analyze

<table>
<thead>
<tr>
<th>Analyze</th>
<th>Angle</th>
<th>Distance</th>
<th>Dynamic</th>
<th>Properties</th>
</tr>
</thead>
</table>

Surface Modeling Improvements

Change Normal
This function supports the new color display of the non-normal side of the surface in shading. The display updates as you select surfaces to flip.

Set Normal
The Set Normal function now displays a selectable arrow for flipping individual surfaces. This function supports the new color display of the non-normal side of the surface in shading.

Orient Normals
This new function positions the normals of neighboring surfaces to one another. Once you set the initial position, you can flip the results and display arrows that show the direction. You can also manually select any of the arrows or surfaces to flip individual surfaces. Like the other Edit functions, Orient Normals supports the new color display of the non-normal side of the surface in shading.

Surface to Surface and Surface to Plane Auto Analyze
In X8, the Fillet Surface to Surface and Surface to Plane functions display a Maximum Recommended Radius. Once you select the surfaces, Mastercam analyzes them and displays a maximum recommended radius that can be successfully created.
Mill Enhancements

2D Milling

2D HST Improvements

- 2D HST now supports multi-core processing on toolpaths with multiple machining regions.
- Consolidated Dynamic Area, Core, and Rest toolpaths down to one single Dynamic Mill toolpath. Also consolidated Area, Core, and Rest Mill toolpaths down to one single Area Mill toolpath.

- 2D HST Dynamic Mill includes first pass feed reduction and offset on the Cut Parameters page. This reduces the feed rate for the first pass of a dynamic mill toolpath entering stock from outside. It also expands the machining region to ensure the tool does not engage a lot of material on the very first pass due to an unknown exact block size.
Region Chaining

- Removed single region chain support to provide more toolpath control.

![Chain Options X7](image1)

![Chain Options X8](image2)
- New Air Region chains define safe air the tool can move within via closed chains.

- Open Air Region chain support defines an open side of a closed machining region or closes an open machining region chain.
- New Containment chains constrain machining region motion.
- Job setup stock shape support used when no machining region is selected or open machining region chains are selected. Open machining region chains are extended to block tangent edges or shortest distance.

- When no stock is defined and no open air is defined, open machining chains are closed with a straight line.
2D Contour Enhancements

You can now adjust feeds and speeds on the Multi Passes page, making it easier to control your tool motion. In addition, Mastercam X8 adds corner rounding support in Ramp and Oscillate modes.
**3D Milling**

**Z Stepdowns**

3D high speed toolpaths based on Z-level cuts now process Z stepdowns across multiple cores.
3D HST Project Depth Cuts

Mastercam X8 includes new support on Cut Parameters page for multiple depth cuts. The **Order by depth** checkbox controls the cut order by depth or by input entity.

Tilt Away to Avoid Collision

These new options in 3D HST finishing toolpaths provide an easy-to-use introduction to 5-axis technology. Additional options on the Holder page use tilting to avoid shank and holder collisions during multiaxis motion. This toolpath requires a spherical tool.
NOTE: These options require a Mastercam Multiaxis license.
Multiaxis Milling

Multiaxis Roughing
Multiaxis Roughing is a new strategy added for X8. It allows the easy creation of pocketing toolpaths to rough out pockets that have a curved floor surface. It supports offset or Dynamic style strategies, and multiple depth floors.

Safety Zone Improvements
Improvements to the Safety Zone feature in multiaxis toolpaths makes defining and editing the safety zone much easier. On the dialog box, seeing a representation of the shape while defining makes it much easier and more precise to enclose the targeted volume.
Tool Inspection

Tool inspection was added to many of the multiaxis toolpaths, including Blade Expert and Port Expert. After you enter an inspection parameter, either in distance or time,
the tool will retract and trigger an inspection code at the next linking move. Your post processor must be modified to support this new feature.

![Screen capture of MASTERCAM X8 Mill Enhancements](image)

### Additional Improvements
- Enhancements to oscillation in Curve 5-axis.
- Ability to finish blades in Blade Expert with a SWARF algorithm.

### Mill Tooling

#### Tool Creation and Editing
The old style Define Tool dialog box has been replaced with the new interface throughout the product. This provides a more consistent user experience for creating and editing tools. Some of the existing tool types are also updated so they no longer require custom profiles.
In preparation for future updates, the group boxes include expanders for advanced properties.

**Edit Holder**

The Edit Holder interface has been updated to support additional properties. Prior to X8, all non-geometric attributes had to be edited using the stand-alone Tool Manager. With the X8 release, you can edit properties such as the upper and lower connection type and size from within Mastercam.

The right-click menu in the Mastercam Tool Manager includes a new **Edit Holder** option. Selecting this option displays the edit holder interface.
The **Edit Holder** option is still available in the Holder parameters page for tree-style toolpath dialogs, but making it available in the Mastercam Tool Manager is useful when working with older toolpath types that do not offer the Holder page.

In addition, the .TOOLDB format is now the standard for holder libraries. This replaces the .HOLDERS file format.

**Edit Assembly Names**

Changing tool, holder, and assembly names is now available through a right-click menu option in the Mill operation tool parameters page.
The Assembly Name field has also been added to the Holder page for convenience.
Calculate Feeds and Speeds
The calculate feed/speed button has been added to the Create and Edit tool dialog boxes. This option is useful when you are working on tools inside of Mastercam because it can query the workpiece material of the active machine group.
Standalone Tool Manager
Geometry properties have been removed from the property grid and replaced with a button that will launch the Edit Tool dialog box. You can also double-click on an item in the data grid to edit the tool.

Productivity+ Enhancements

- Update to Renishaw Productivity+ Version 2.0.
- Support for extended character languages.
- Added graphic selection for Calibration Point position.
- Probe posting dialog box now resizable.
Updated Configuration Tool now includes kinematics configuration interface.
Lathe Enhancements

*Lathe Stock Model Support*

Lathe toolpaths can now be included in stock model operations, which can be used for subsequent milling operations.
New Contour Roughing Toolpath
This new roughing toolpath is designed to take passes parallel to chained geometry. This toolpath is useful for parts where the initial stock shape is similar to the final part shape.

Additional Improvements
- **Added Incremental Depth Cuts** to Lathe Rough toolpath that allow incremented and decremented depth cuts.
- Disable the **Tool Display** option to see significant processing speed improvements for Dynamic Lathe and other toolpaths.
- Lathe stock, tailstock, and steady rest are now associated with any solid body used to define them.
- New upper and lower profile options in Create Boundary are very helpful for Lathe users.
Mill-Turn Enhancements

**Level 1 Now Available**
This new option for Mastercam Mill-Turn is perfect for customers with twin turret machines and B-axis solutions that do not require surface machining or 5-axis toolpath options.

**Reference Positions**
Mill-Turn now includes the ability to define and select reference positions for the approach and retract moves in operations. This provides much better control for repositioning between operations. Common reference positions are pre-defined in the machine definition, allowing for reuse in each job. Job-specific reference positions are also easily defined through a dialog accessed from the Toolpaths Manager.
Selecting reference positions to use at specific times during the toolpaths occurs in the Sync Manager. The settings are reflected in the Sync Manager graphical display and in simulation.

**Wibu Licensing**

Mill-Turn .MACHINE files now require WIBU licenses and are secured binary files. Third-party development of .MACHINE environments is now possible through Devel-oper licenses.

**Job Setup Enhancements**

- Pickoff/cutoff options are now available for single pieces of stock in both left and right spindles. Machine definition now controls the availability of
continuous bar stock options based upon settings in the chuck component (for example, if the chuck is plugged, you cannot use bar stock).

- Mill-Turn now checks the defined chuck bore diameter against the stock diameter size. If you select a **Continuous bar stock** option, error messages appear when the stock is larger than the bore. If you select a **Single pieces of stock** option, the stock must be positioned beyond the face of the chuck if the stock diameter is larger than the bore.
**Tool Loading Enhancements**

- Tool Setup Manager now accessible from a Toolpaths Manager button.
- Added ability to filter tools by Active Turret/Spindle.
- Drag-and-drop ability added for placing tools on locators loaded on turrets.
- Right-click menu option in operation parameter dialog now allows you to load tool and view machine.
- Added ability to load tools that are not used in operations but present in machine. This allows for collision detecting tools that are present in the machine turret, but not actually used in operations.

**Sync Manager Enhancements**

- Connected Stream List Display toggle added. This allows for traditional locked scrolling and linked operation branch display versus independent control by clicking the toggle or selecting a right-click menu option.
MACHINE PROPERTY CHANGES

- Mastercam file link indicator moved from Code Expert status bar to document tab, which makes it easier to see the link status.

**Machine Property Changes**

- NC Output Settings now available for each machine.
- Default NC Destination and Overwrite mode moved from Application Options - used for all operations in Mill-Turn Machine Group in Toolpaths Manager, but editable for each job in Machine Group Properties.
- Stream control added to allow for file name customization for each stream.

![Machine Configuration and NC Output Settings](image)

**Additional Enhancements**

- Turret Park operations have been changed to select the turret/head to move and the destination. This change was required to support machines with multiple turrets in the same stream.
- Twin turret machines such as the Okuma LU Series, Doosan PUMA TT Series, and others are now supported.
- Added Manual Entry operation support to Mill-Turn, allowing users to enter comments and code as needed.
Attention! Updates may be available.  
Go to Mastercam.com/Support for the latest downloads.